Proceedings of Indian Geotechnical Conference
December 15-17, 2011, Kochi (Paper No.N-281)

FINITE ELEMENT MODELING OF GROUND - STRUCTURE INTERACTION CONSIDERING NON-LINEAR RESPONSE OF THE GROUND

F. Jancy, M. Tech. Student, IIT Hyderabad, ce10m04@iith.ac.in
A. Rajagopal, Assistant Professor, IIT Hyderabad, rajagopal@iith.ac.in
B. Umashankar, Assistant Professor, IIT Hyderabad, buma@iith.ac.in
M. R. Madhav, Visiting Professor, IIT Hyderabad, madhavmr@gmail.com

ABSTRACT: Response of the ground on which the structure rests will have a bearing on the distribution of forces in the structural members. Conventional methods of structural analysis and design assume often fixed bases for various loading conditions. A realistic analysis and design procedure should include actual support flexibility, nonlinear and heterogeneous nature of the soil together with nonlinear soil-structure interaction effects. Such an analysis would result in overall stiffness of the soil-foundation-structure system, realistic to the existing conditions. This work focuses on the computational modeling of ground-structure interaction using finite element package ANSYS. To demonstrate the behavior of structure while considering actual nature of ground response, a simple portal frame is analyzed. Portal frame is modeled as linear elastic, whereas the ground is modeled as both linear elastic and non-linear elastic-plastic behavior. The study gives insight into variation of displacement of portal frame while considering linear and non-linear behavior of ground.

INTRODUCTION

The term soil-structure interaction has been largely used for mechanics of interaction between soil and the structure or its part embedded in it. Numerous studies are available in the literature on the effects of soil-structure interaction under various loading conditions [1, 2, 3, 4]. The conventional design of framed structures resting on ground usually involves assumption of fixity at the base of foundation, neglecting the flexibility of the foundation and the deformation response of ground. The foundation settlement can alter the distribution of forces in the framed structure. Hence, assessing the structural response in conjunction with the ground response becomes important.

Complexity in soil-structure interaction problems has several sources, as shown in Fig 1, which include:

a) Material non-linearity of soil, for example elasto-plastic behavior,
b) Material non-linearity of building materials, such as cracking or damage,
c) Geometrical non-linearity of the soil-structure interface,
d) Coupling, since the boundary conditions on the structure (in terms of loading or of displacement) result from an interaction.

In this study, structural response is studied considering material non-linearity of soil as the source of ground-structure interaction problems. The results are discussed in the following sections.

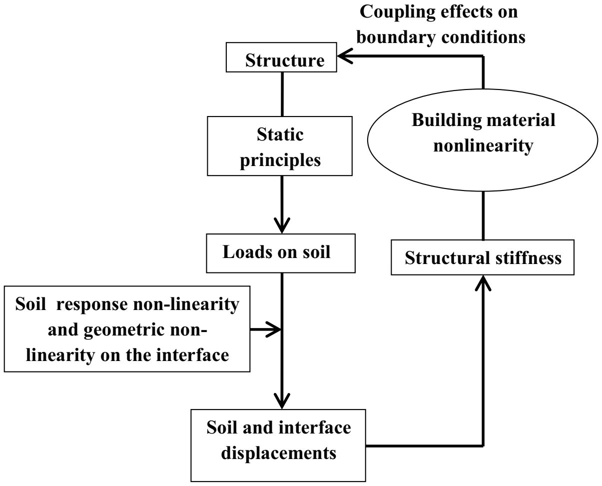

Fig.1 Flow chart for soil-structure interaction [2]

PROBLEM DEFINITION

A single-storey portal frame resting on ground as shown in Fig. 2 is considered for ground-structure interaction analysis.
The following cases are modeled and analyzed:

a) Portal frame fixed at the base, not considering ground response and frame modeled as linear elastic (Case 1)
b) Portal frame resting on ground, both frame and ground are assumed as linear elastic (Case 2)
c) Portal frame resting on ground, frame is modeled as linear elastic and soil as non-linear elastic-perfectly plastic (Case 3)

The finite element program ANSYS version 12.1 is used in this study. All the cases are analyzed as 2D problems for a given uniformly distributed load acting on frame and the displacements are compared.

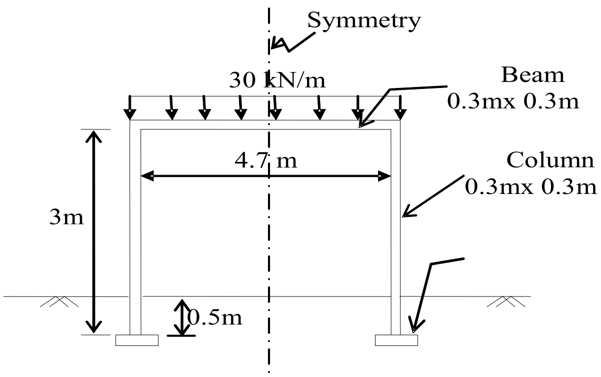

Fig. 2 Schematic of a single-storey portal frame resting on ground

FINITE ELEMENT MODEL

For Case 1, BEAM 3 (2D elastic beam) is used (Fig. 3). BEAM3 is a uniaxial element with tension, compression and bending capabilities. The element has three degrees of freedom at each node - translation in the nodal x and y directions (U_x and U_y) and rotation about the nodal z axis (θ_z).

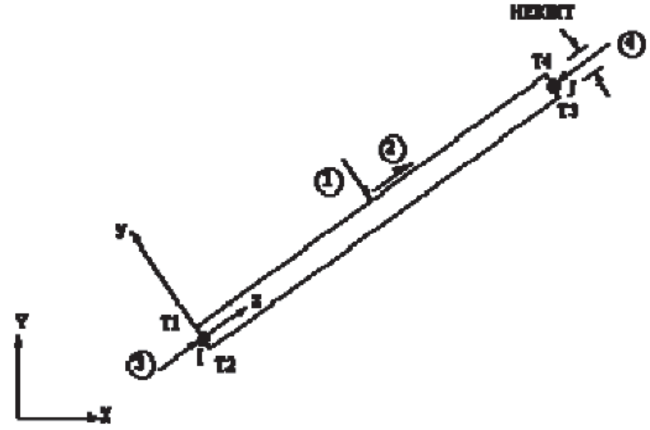

Fig. 3 Geometry of BEAM3 finite element

The 2D Plane 183 element (Fig. 4) is used for modeling of other two cases (Case 2 and Case 3). Plane 183 is a higher order 2D element defined by 8 nodes having two degrees of freedom at each node – translations in the nodal x and y directions. For Case 3, elastic-plastic behavior of the ground is modeled using Drucker-Prager criterion. A brief background on the Drucker-Prager model is outlined in next section.

The properties of ground and frame are provided in Tables 1 and 2 respectively.

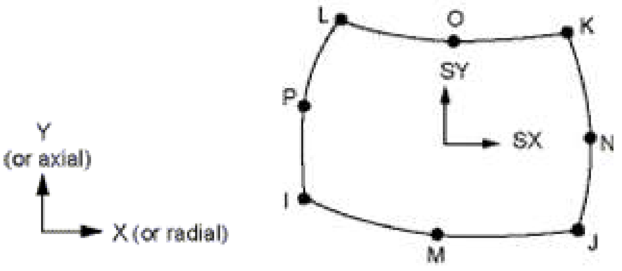

Fig. 4 Geometry of PLANE 183 finite element

Table 1 Soil properties used in modeling

Property	Value
Cohesion	0
Friction angle	36 degrees
Dilation angle	12 degrees
Young's modulus	10 MPa
Poisson's ratio	0.3
Saturated unit weight	18 kN/ m^3

Table 2 Material properties of frame

Property	Value
Grade of concrete	M25
Young's modulus	25000 MPa
Poisson's ratio	0.15

MATERIAL MODEL

The representation of Drucker-Prager perfectly plastic model is shown in Fig. 5 [5]. The failure criterion for the Drucker-Prager model for sandy soils is of the form

$$F = \alpha I_1 + \sqrt{J_2} - k \qquad (1)$$

where I_1 is the first invariant of stress tensor, J_2 is the second invariant of the deviatoric tensor, and α, k are material constants expressed in terms of the well-known shear strength parameters of soil c and ϕ.

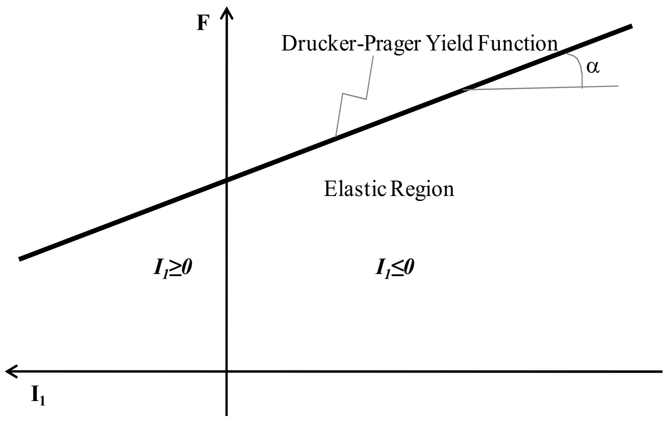

Fig. 5 Representation of Drucker- Prager yield function

DISCRETIZATION

The 2D finite element model for analysis of linear and non-linear problem is shown in Fig. 6. For Case 1, element size is taken as 0.1_m. Taking symmetric conditions, only one-half of the frame - ground model is considered for analysis for Case 2 and Case 3. Model is discretized using mapped meshing technique. Rectangular elements are used to mesh the frame and soil continuum. The element size are decided after performing a number of initial trials with different sizes of meshes of increasing refinement until the displacements did not change significantly with further refinement. Fine mesh is used near to the foundation._For Cases 2 and 3, size of finite elements varied from 0.1_m near the base of the footing to 1_m near the left boundary (Fig. 6). For frame, element size of 0.1 m is taken. The distances to boundaries shown in Fig. 6 ensure semi-infinite medium.

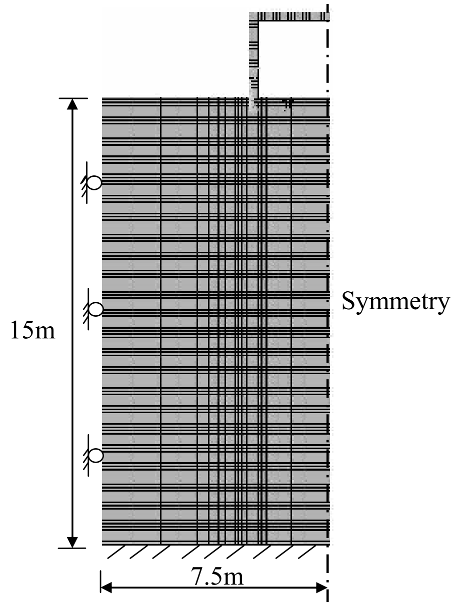

Fig. 6 Finite element meshing for Case 2 and Case 3

ANALYSIS OF SOIL STRUCTURE INTERACTION

All numerical computations of elements are performed using reduced numerical integration. The degree of freedom on lateral boundaries is restrained from moving in the lateral direction. Bottom boundary is restrained from all degrees of freedom. A uniformly distributed load of 30kN/m is applied for all the three cases as shown in Fig. 2. The analysis is performed using Newton-Raphson scheme. The external load is applied in small increments and convergence is checked for several iterations to satisfy the systems equilibrium. The iterations are continued at each load step until the norms of out-of-balance force and the incremental displacements are less than 5%.

RESULTS

The maximum displacement of the center of portal frame is compared for the three cases for an applied load of 30 kN/m. Figs. 7, 8 and 9 show the contour plots of displacement for the three cases considered in this study. Maximum displacements of 5.7 mm, 6.8 mm, and 11.9 mm are obtained for Cases 1, 2 and 3, respectively. The maximum displacement of portal frame in Case 3 (Portal frame modeled as linear elastic and ground as elastic-plastic) is about 109% and 19% higher than that of Case 1 (Portal frame assumed linear elastic and fixed at its base) and Case 2 (Portal frame and ground both modeled as linear elastic), respectively

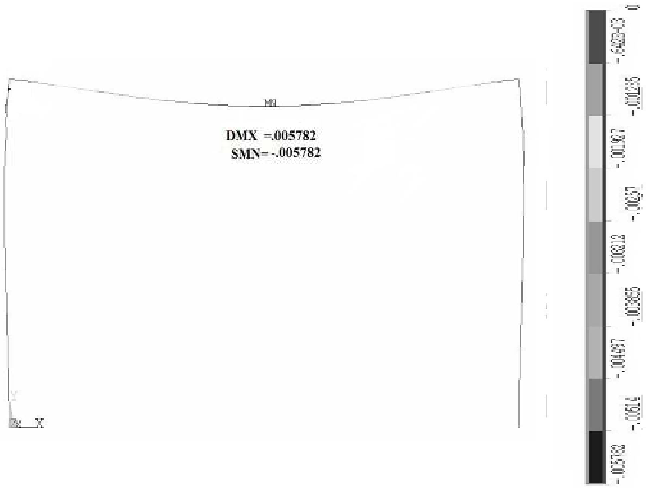

Fig. 7 Contour plot of displacement for Case 1

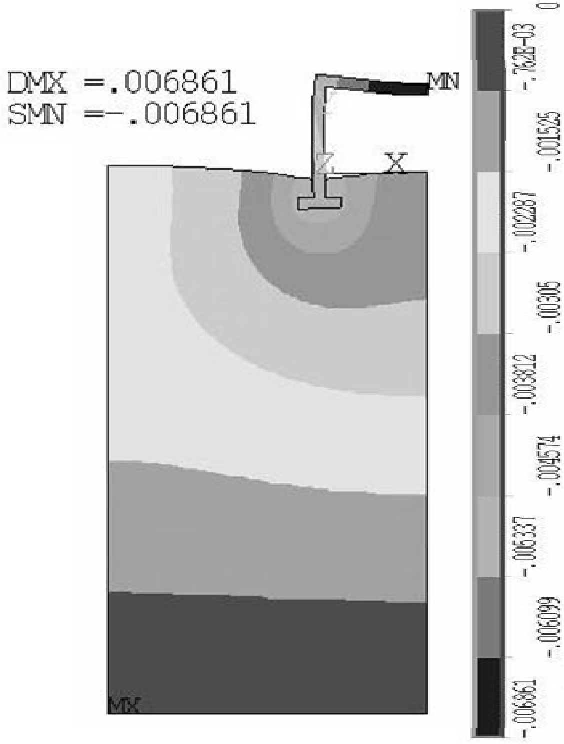

Fig. 8 Contour of displacement for Case 2

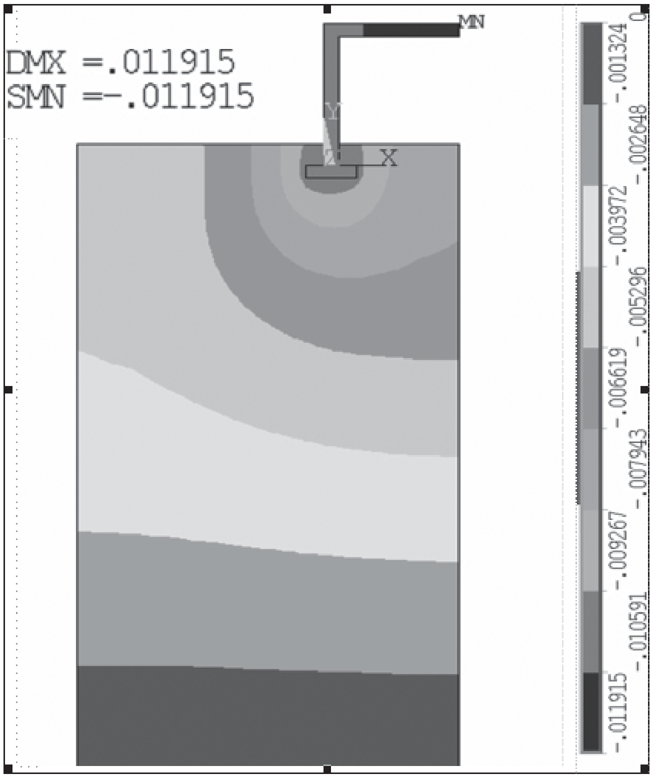

Fig. 9 Contour plot of displacement for Case 3

CONCLUSIONS

From these results, the maximum displacement of the portal frame is higher when the ground interaction was considered than that of fixed base condition. It increases from 5.7 mm to 11.9 mm. For multistoried buildings, ground interaction effect on the portal frame behavior may be more significant and hence, the ground-structure interaction should be accounted for in the structural analysis of important buildings.

REFERENCES

1. Dutta, S. C. and Roy, R. (2002), A critical review on idealisation and modelling for interaction among soil-foundation – structure system, *Computers and Structures* (80), 1579-1594
2. Breysse, D., Niandou, H., Elachachi, S. and Houy, L. (2004), A generic approach to soil-structure interaction considering the effects of soil heterogeneity, *Geotechnique* 54, No. 2, 143-150
3. Houy, L., Breysse, L. and Denis, A. (2005), Influence of soil heterogeneity on load redistribution and settlement of a hyperstatic three-support frame, *Geotechnique* 55, No. 2, 163-170.
4. Karthigeyan, S., Ramakrishna, V.V.G.S.T. and Rajagopal, K. (2007), Numerical investigation of the effect of vertical load on the lateral response of piles, *Journal of Geotech. and Geoen. Engineering*, ASCE , 133(5), 512-521
5. Yu, Hai-Sui (2006), *Plasticity and Geotechnics*, Springer Publisher, Ed. 1
6. *ANSYS reference manual*